

FIRESTRUC – Integrating Advanced Three-dimensional Modelling Methodologies for Predicting Thermo-mechanical Behaviour of Steel and Composite Structures Subjected to Natural Fires

STEPHEN WELCH¹, STEWART MILES², SURESH KUMAR³, TONY LEMAIRE⁴
and ALAN CHAN¹

1 – School of Eng. & Elec., University of Edinburgh, Edinburgh, EH9 3JL, UK

2 – Formerly, BRE, Bucknalls Lane, Garston, Watford, WD25 9XX, UK

3 – BRE, Bucknalls Lane, Garston, Watford, WD25 9XX, UK

4 – TNO Efectis, 6 Van Mourik Broekmanweg, NL-2628 XE, Delft, Netherlands

ABSTRACT

A hierarchy of coupling strategies for integrating advanced three-dimensional modelling methodologies for prediction of the thermo-mechanical response of structures in fire has been developed and systematically assessed. The methods have been implemented to link various fire and structural code pairs and a common file format has been established for one-way coupling, facilitating addition of further codes in future. Methodologies based on one-way coupling, with information passing from fire to solid-phase code, have an advantage of flexibility, particularly where omission of structural components from the fire model can be justified, i.e. where there is only a minor influence on the overall gas temperatures and radiative exchange. Where structural components markedly affect the flowfield there is no alternative to including them in the fire simulation, though geometrically simplified representations, using adiabatic solids, may be sufficient. Two-way coupling should be considered when there is a more significant interaction of fire and structure, or when more accurate analysis is required and greater computational costs can be sustained.

KEYWORDS: CFD, FEM, solid phase, modelling, coupling

INTRODUCTION

Effective coupling of detailed model predictions of fire conditions and the associated thermal and mechanical response of a structure exposed to a fire would potentially provide a far more flexible and general means of analysing structures in fire than has traditionally been used. Linked Computational Fluid Dynamics (CFD) and Finite Element Method (FEM) codes are already exploited quite routinely in some areas of science and engineering, e.g. blood flow analysis, but thus far relatively little for fire applications. In particular, whilst some progress has been made for individual code pairs [1–5], and in commercial software (e.g. ANSYS-CFX Fluid structure interaction (FSI)/MFX multi-field solver), little serious effort has been made to establish generic and transparent linkage mechanisms and there have been no previous systematic studies of the methodologies for achieving coupling. This work, part of a European Community project funded by the Research Fund for Coal and Steel (RFCS), known as FIRESTRUC [6], set out to address these issues with a broad examination of approaches to coupling, considering the implications for accuracy and the computational demands. In order to facilitate the study, different types of couplings were established between a number of fire and structure code pairs, and these were exercised in application to a number of test cases, both hypothetical and experimentally based.

COUPLING APPROACHES

The FIRESTRUC project focused on the development and assessment of alternative methodologies to integrate the capabilities provided by models of fluid dynamics and thermal radiation, and solid-phase thermal and mechanical response, in fire. The discrete capabilities (calculations), which collectively address these issues, can be broken down as follows:

- Calculation of the gas temperatures, velocities, chemical species, etc., within the fluid domain of a building geometry in the presence of a given fire source.
- Calculation of the associated radiation field (radiance values, incident fluxes, etc.).

- Calculation of the thermal boundary conditions where the solid (building) elements adjoin the gas phase, including surface temperature, incident (from fluid to solid) radiative fluxes and convective fluxes (fluid to/from solid).
- Calculation of the thermal response (temperature rise) within the solid elements.
- Calculation of the associated mechanical response of the solid elements.

CFD codes were used for the fire simulation, whilst both FEM and finite volume codes were adopted for the structural modelling. The latter is therefore strictly referred to as “solid-phase”, though FEM is also used as a short-hand form. The calculations of these codes may be considered independently, loosely coupled to one another, or tightly coupled into (in effect) a single calculation. Typically, the CFD models address the first two items and the solid-phase models the last two; if coupled codes are not available, then approximations are required for the third item, since the boundary condition is a function both of the fire and the structural response. This coupling of fire and solid-phase models is far from straightforward and different levels can be defined, varying significantly in complexity. In overview, predicting the full thermo-mechanical response of a fire-exposed structure generally involves three main steps (see Fig. 1):

1. Simulation of the fire within the compartment or building.
2. Simulation of the heat penetration into the structure, i.e. the thermal response.
3. Simulation of the mechanical response of the structure, given its thermal response.

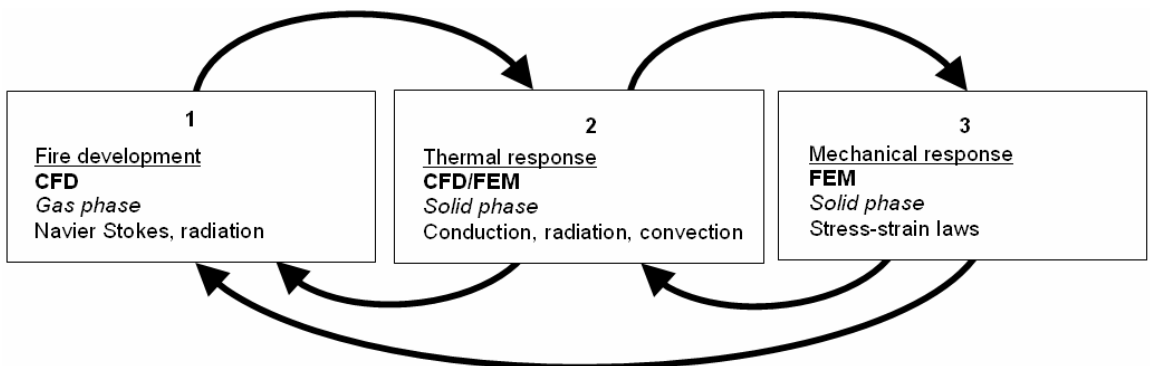


Fig. 1. Coupling methods for CFD and solid-phase (FEM) simulations of fire-exposed structures.

Conventionally, step 1 is carried out with CFD (typically, a CFD fire model includes a radiation sub-model, and the CFD and thermal radiation calculations are performed as an integrated system), step 2 can be addressed either with CFD or solid-phase codes, and step 3 is undertaken entirely by the solid-phase code. In order to connect, or “couple”, the fire and structural parts of the simulation, boundary condition data must at some point be exchanged between the CFD and solid-phase codes. However, this process is complex and often compromised by the inherent imperfections, e.g. it may not be practically possible to achieve a good match in the definition of the relevant structural components between the fire model, where resolving the full geometrical details can be excessively demanding, and solid-phase model, which is generally free of such constraints. Also, real-time modification of CFD geometries, as they are deformed, is not generally possible. Hence, it is inevitable that many different levels and approximations are defined in practice. Three main stages of coupling can be distinguished:

- Coupling between steps 1 & 2, i.e., gas temperatures, incident radiative heat fluxes and convective properties are transferred from the fire development (gas phase) calculation to the thermal response (solid-phase) calculation for the solid, and *vice versa*, surface temperatures calculated by the thermal response calculation are transferred back to the fire development calculation.
- Coupling between steps 2 & 3, i.e., temperature distributions within the structural elements are transferred from the thermal response calculation to the mechanical response calculation, and *vice versa*, the effect of deformations such as cracks can be transferred back.
- Coupling between steps 1, 2 & 3, e.g., the effect of deflections obtained from the mechanical response calculation can be considered on the gas flow predictions for fire development.

As mentioned above, coupling is affected by scale, and it is important to note that the spatial scales of the thermal response models are typically at least an order of magnitude smaller than those of the fire development and mechanical response. Also, there are differences in the characteristic time scales of the various modelled processes. Furthermore, time and space need to be discretised to solve the problem in both CFD and solid phase, but the discretisation for the three steps generally need not coincide, and, for the simulation of the fire development, a 3D analysis is required, whereas for the thermal response a 1D or 2D approach will often suffice.

Coupling fire development and thermal response

The interaction between the fire development model (CFD) and the thermal response model (CFD or FEM) can be approximated at different levels of detail, as discussed below.

One-way versus two-way interaction

Although the interaction between the fire and the thermal response is fundamentally two-way, a simple one-way coupling may be advantageous under certain conditions. In the latter approach the fire development is calculated in a manner entirely independent of the thermal response, based on pre-defined boundary conditions. From the fire development model, the gas temperature, convective flow characteristics and radiance values/heat fluxes, all varying in time and space, are transferred to the thermal response model. Interpolation of these data between the different temporal and spatial scales used by the models is required. Thereafter, the thermal response model calculates the temperature distribution in the structural elements on the basis of the interpolated output of the fire development model. Because the thermal response of the structure is not directly represented in the fire simulation the structural components may optionally be completely omitted from the fire model (see discussion below).

In the two-way approach the surface temperature of the structure is calculated taking the in-depth heat penetration into account, using one or more thermal response models, and transferred back to the fire development model. In this case it is a requirement that the structural geometry is explicitly represented in the fire model, but it will typically require less detail than in the thermal response model.

Including structural geometry in fire model

As mentioned above, with one-way coupling the structural component may optionally be entirely omitted from the fire development model. This might be an appropriate strategy when the component is relatively small compared to the overall volume of the computational domain, e.g. when dealing with I-section columns, but care must be taken with elements such as deep structural beams which may have a disproportionate effect on the thermal fields due to interference with ceiling jets and formation of smoke reservoirs. For clarity, a modification of Fig. 1 is shown below in Fig. 2, where the thermal and mechanical responses are combined but a distinction is now made between omitting or including the structural component in the fire model. When omitted, the coupling is purely one-way; when included, one and two-way coupling are represented by return paths C and D respectively.

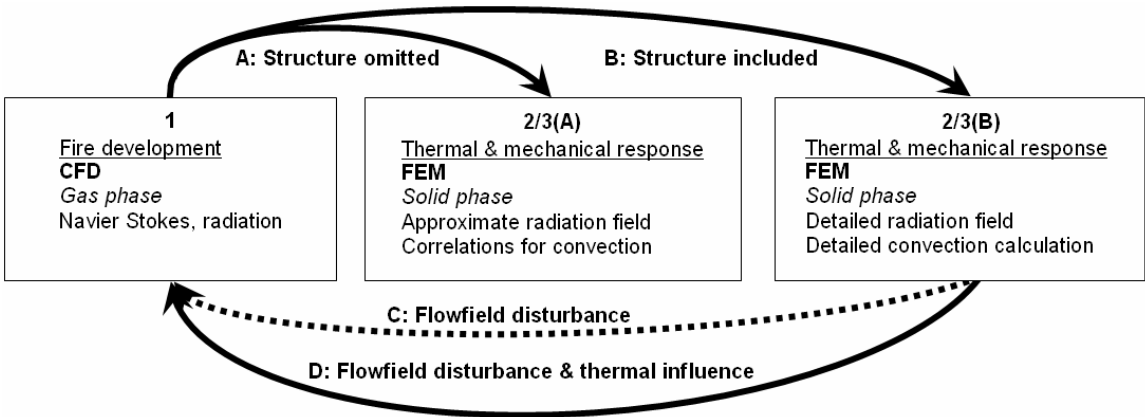


Fig. 2. Coupling methods for CFD and solid-phase (FEM) simulations of fire-exposed structures.

Merged versus segregated solver

The most tight interaction between the fire development model and the thermal response model(s) is a complete merger of the systems of equations of each. In this case, both the fire development and the thermal response are solved within the CFD software, an approach known as “conjugate heat transfer”. A disadvantage however is the increase of the size of the resulting system of equations, especially in the case that the thermal response model(s) are using a fine grid.

Alternatively, the systems of equations remain separated and calculated data is exchanged at the boundaries. Obviously, this segregated approach is also followed when the fire development is calculated in CFD and the thermal response in an independent solid-phase code. The data can be exchanged after each iteration or after each timestep or even after every few timesteps. The level of detail in the information exchanged depends on the respective spatial discretisations, i.e. of the compartment in the fire development model and of the structure in the thermal response model(s).

Simulations for representative parts of structure

As the spatial scale of the model for thermal response is generally much smaller than that of the fire development, separate simulations are usually made for representative parts of the structure, e.g., beams and columns, rather than one simulation of the heat penetration of the entire construction.

Coupling thermal response and mechanical response

Different levels of coupling can again be distinguished. The basic case is a one-way interaction in which the temperature distribution calculated in the thermal response model(s) is used as an input for the mechanical response model. If the data is only transferred from the thermal response to the mechanical response without a reverse process, the thermal response can be calculated over the entire time domain and transferred only once at the beginning of the mechanical response simulation. A more advanced, two-way interaction includes the effect of deformations and displacements on the thermal response models. For instance, the calculation of heat leakage through cracks and gaps that arise during fire requires such a two-way coupling. However, this type of coupling would typically be highly uncertain, given the great difficulty of predicting the opening of cracks and gaps during fire in any reliable or accurate way. Moreover, experimental data regarding the effect of crack width and gap openings on the thermal response is scarce. Therefore, only a one-way coupling of the thermal response to the mechanical response has thus far been considered. The biggest challenge here is to overcome the scale difference between the respective phenomena, since the mechanical response will often be calculated over much larger scales, similar to those of the fire development.

Coupling mechanical response and fire development

A deflection of a roof and compartmentation failure might potentially modify the flow of the hot gases and smoke [3]. In order to take this effect into account, the CFD grid must be stretched on the basis of the nodal displacements. However, not all CFD codes are capable of dealing with non-Cartesian grids. Moreover, a modification of the mechanical properties of the structure in the design process would require a new simulation of the fire development, rendering such an approach generally unsuitable for practical design problems. Nevertheless, for CFD codes that can handle non-Cartesian grids, an interface could in principle be made, matching the nodes from the mechanical response model to the nodes of the fire development model. An update of shape of the CFD grid with the thermal response bodies could be carried out at appropriate intervals. The mechanical response model would need to be updated with the temperature data accordingly. For the more general case of integrity failure, the phenomena are even more complex and uncertain; here, neither of these feedback mechanisms from structure to fire were considered further.

METHODOLOGY HIERARCHY

The study of the coupling methodologies has focused on the implementation and testing of a representative set of integration approaches, in a hierarchy labeled A – F, as summarised below. In all cases, information is converted by the CFD and solid-phase/FEM codes, and exchanged between them. Methods differ in the type of data exchanged and thus in the kind of data conversion performed by each code.

Two-way coupling (Methods A – B)

Tight coupling is achieved when the models of fire development and thermal response are linked in a two-way segregated approach, exemplified by methods A and B. These methods are based on the use of pre-calculated conversion factors for the conversion of the thermal data. They are quite similar, but in Method A, effort on interpolations and radiation calculations is shared between the CFD and FEM codes, whereas all the work is done in the CFD code in Method B. The implementation of Method B is described as follows (see Fig. 3):

1. The CFD code generates a file with a description of the surface mesh of the CFD solids in a specified format. The surface mesh is made up of so-called CFD faces, with each face representing one mesh node. Each face is described by the position of its vertices.
2. The FEM code generates a file with a description of the surface mesh of the FEM solids in the same format as used to describe the CFD solids.
3. A program called CTINP computes the required conversion factors for the CFD code. The conversion factors (interpolation factors and view factors) are written to a file in a specified format.
4. On initialization, the CFD code reads the file with the conversion factors.
5. The CFD code computes, after each CFD iteration, the values of the environmental temperature, $T_{env,j}$, and environmental heat transfer coefficient, $\alpha_{env,j}$, for each surface mesh node j of the FEM solid. The calculation is based on the gas temperature, $T_{g,i}$, the convective heat transfer coefficient, $\alpha_{c,i}$, and the incident radiation flux, $q_{g,rad,i}$, on each surface mesh node i of the CFD solid using the pre-calculated conversion factors. The values are written to a file in a specified format.
6. The FEM code reads the values of $T_{env,j}$ and $\alpha_{env,j}$ before each FEM iteration and applies these values as boundary conditions for the temperature computation.
7. After each FEM iteration the FEM code computes the values of the surface temperatures, $T_{s,i}$, for the CFD solid based on the conversion factors. The values are written to a file in a specified format. The values of the emission coefficients of the surfaces are also written to the file to account for temperature-dependent emission coefficients calculated by the FEM code.
8. The CFD code reads the values of the surface temperatures, $T_{s,i}$, on the CFD mesh before each CFD iteration and applies these values as boundary conditions for the flow computation.

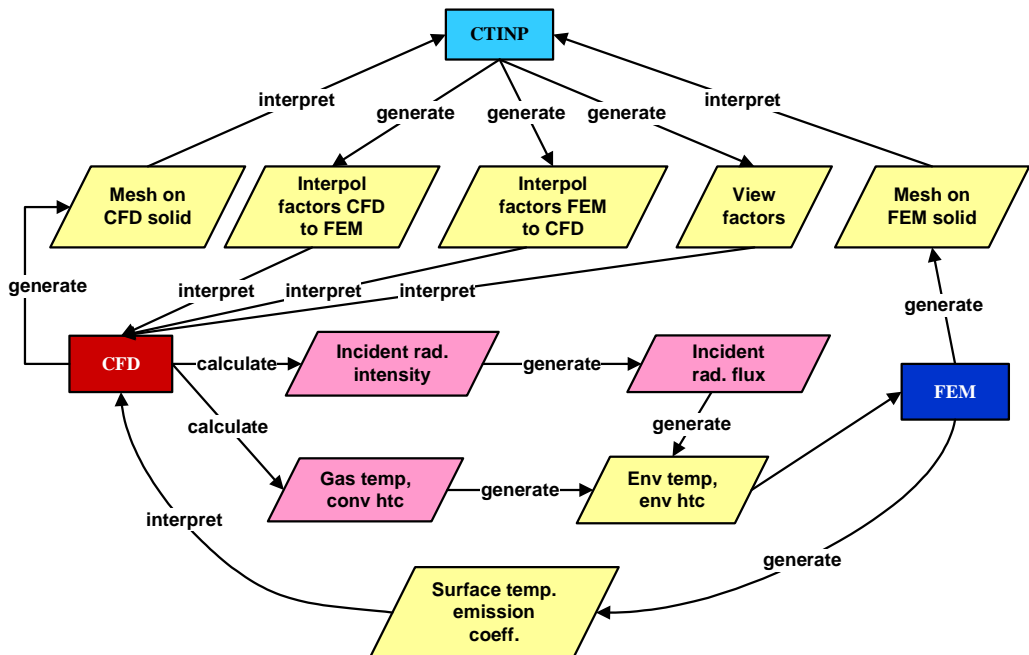


Fig. 3. Method B diagram.

The surface mesh of a CFD or FEM solid consists of faces. Each face represents the surface belonging to one mesh node i (CFD mesh) or one mesh node j (FEM mesh). For convex FEM surfaces the interpolation factor $G_{j,i}$ (from face i) of the total convective heat transfer to face, j , is actually determined by the projection of the area of face i on face j . The following basic procedure is followed to achieve this:

1. The FEM-face j is sub-divided into small sub-faces. On those sub-faces the vertex normal vectors are interpolated. The normal vectors of the sub-faces are extended and for each CFD-face i is determined if it is intersected by the extended normal vector.
2. If one CFD-face is intersected, the area of the sub-face (FEM) is assumed to interact with it alone.
3. If there are more CFD-faces intersected (actually, edges or vertices), the area of the FEM sub-face is divided by the number of intersected CFD-faces, which gives the interaction area for each of the CFD-faces. Hence, go back to step 1 (sub-divide further) until convergence is achieved.
4. Finally, the conversion factors are constructed by dividing by the total area of the FEM-face. This results in a series of conversion factors for a FEM-surface to various CFD-faces, which sum to unity.

Note: The values of $T_{env,j}$ and $\alpha_{env,j}$ depend also on $T_{s,j}$, due to the radiation emitted by the surface (even for a concave solid with no shadow effect). Inner iteration loops in the FEM computation are therefore required to update $T_{s,j}$ for a given set of $T_{g,j}$, $\alpha_{c,j}$ and $q_{g,rad,j}$. These inner iterations may be left out, as done in Method B, if the change of $T_{s,j}$ can be considered slow compared to the change of the thermal flow conditions.

One-way coupling (Methods C – F)

Loose coupling is achieved when the models for fire development and thermal response are linked in a one-way segregated approach. Method C is a subset of Method A, but without feedback to the gas-phase. It consists of the following steps (see Fig. 4):

1. The fire development is calculated by the CFD code neglecting the structural elements and assuming certain thermal boundary conditions for walls, ceiling and floor.
2. In the FEM code each structural element is modelled as a “FEM profile”. A FEM profile may consist of several segments with the same cross-section connected to each other along the axis of the profile. Each segment is assumed to have infinite conductivity and therefore a single uniform temperature.
3. The geometric description of all FEM profiles is written to a file. The description includes the specific location of the FEM mesh nodes on the axis of each profile, the orientation of each profile and the shape of the cross-section of each profile. It also contains the absorptivity of the elements.
4. The CFD code (or CFD postprocessor) reads the file with geometric data made in step 3 and converts the thermal results of the whole (time-dependent) CFD run to the specific locations of the mesh-nodes (segments) of each FEM-profile. The converted thermal results consist of $T_{g,j}$, $\alpha_{c,j}$, and $q_{g,rad,j}$.
5. The FEM code reads the file with thermal data made in step 4 and computes the time-dependent temperature/mechanical response using this data as a boundary condition.

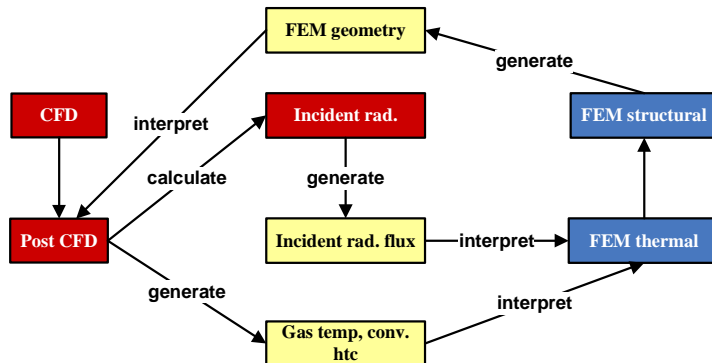


Fig. 4. Method C diagram (Incident Rad. = Incident radiation intensities).

Method D is an extension of Method C and is also based on the use of FEM profiles. Instead of transferring the thermal data from the CFD code to the FEM code using formatted ASCII files, the temperature of the element is computed by a simple thermal module added to the CFD code (or CFD postprocessor). As a consequence the temperatures of the structural element are directly fed into the structural part of the FEM model. In addition to the geometry and position of the structural elements, information about the material properties of the elements is also required.

Method E is similar to Method C, but instead of computing the incident radiative fluxes, $q_{g,rad,j}$, in the CFD code, this is done in the FEM code using the directional radiance values, $I_{k,j}$, on the mesh-nodes j as delivered by the CFD code. Transfer of shape and orientation of the FEM profiles to the CFD code is therefore not required. The method consists of the following steps (see Fig. 5):

1. The fire development is calculated by the CFD code neglecting the structural elements and assuming certain thermal boundary conditions for walls, ceiling and floor.
2. In the FEM code each structural element is modelled as a “FEM profile” – when required in more detail as in Method C, i.e. with finite heat conductivity.
3. The specific location of the FEM mesh nodes on the axis of each profile is written to a file as so-called “FEM lines”.
4. The CFD code (or CFD postprocessor) reads the file made in step 3 and converts the thermal results of the whole (time-dependent) CFD run to the specific locations of the mesh-nodes (segments) of each FEM line. The converted thermal results consist of the gas temperature, $T_{g,j}$, the convective heat transfer coefficient, $\alpha_{c,j}$, and directional radiance values, $I_{k,j}$, on each mesh-node j .
5. The FEM code reads the file with thermal data made in step 4 and computes the time-dependent temperature/mechanical response using this data as boundary condition. The FEM code must be able to handle the shadow effect when appropriate.

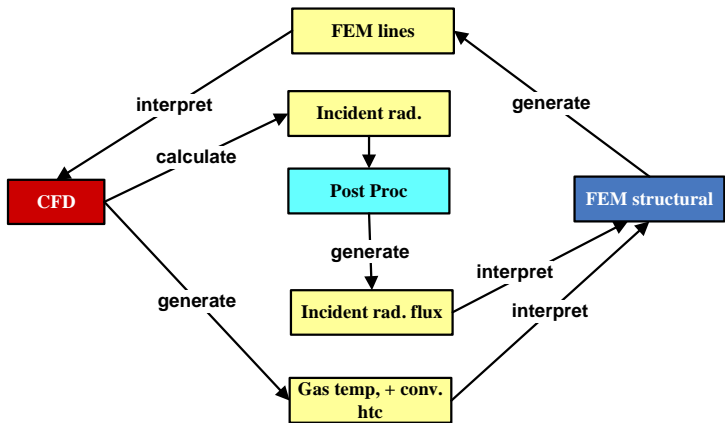


Fig. 5. Method E diagram (module Post Proc could be part of the FEM code).

Note: The amount of thermal data to be transferred to the FEM code is much less in Method C, with the values of just three variables transferred per mesh node, while in Method E the values of $(2 + nk)$ variables have to be transferred, with nk the number of radiance values, $I_{k,j}$, per mesh node. With a typical value for nk of 100 this results in a file which over 30 times bigger than with Method C.

In Method F the same thermal data as in Method E is interpolated on a coarse CFD mesh (not the original mesh) and saved in a data file in a format accessible by the FEM code. Interpolation of thermal data from this coarse mesh to the FEM mesh is carried out entirely by the FEM code. The thermal data consists of the gas temperature, $T_{g,j}$, the convective heat transfer coefficient, $\alpha_{c,j}$, and directional radiance values, $I_{k,j}$, on each coarse CFD mesh-node j . Method F saves data storage memory, because the original CFD data can be deleted after the interpolation is performed. However, if the number of coarse mesh nodes becomes too large it could be more memory efficient to keep the original CFD gas and surface temperatures (deleting the other field variables), recalculating the thermal data when required, either on a coarse CFD mesh (this method) or on the FEM mesh nodes (segments) of FEM profiles (Method C) or FEM lines (Method E).

IMPLEMENTATION

In order to assess coupling methodologies a number of code pair linkages were developed for the software packages routinely used by members of the FIRESTRUC consortium, specifically VESTA, SOFIE, JASMINE and FDS for fire simulations and DIANA, ANSYS, SAFIR and STELA for solid-phase analysis [1,5,6]. Each of these codes has its own requirements for format of inputs and outputs. However, the intention of the study was to assess the coupling methodologies, not to generate new “software products”, and in order to maintain transparency and increase generality a common interface file format for one-way coupling was also defined, as described below.

The specific couplings examined within the project are set out in Table 1 [6]. For pairings, the CFD code is given first, followed by the solid-phase code. The solid-phase codes are all based on the finite element method, with the exception of STELA which uses a finite volume formulation for thermal calculations.

Table 1. Code pairs examined in the FIRESTRUC project for specific coupling types and levels [6].

Type of coupling	Level of coupling		
	One-way	Two-way	
		Segregated	Merged
Fire development to thermal response	JASMINE-SAFIR, SOFIE-SAFIR, VESTA-SAFIR, FDS-ANSYS, JASMINE-STELAb	VESTA-DIANA	JASMINE-STELA
Thermal response to mechanical response	SAFIR, DIANA, ANSYS	None	None

The couplings implemented for JASMINE-SAFIR, SOFIE-SAFIR and VESTA-SAFIR are based on the one-way procedure of Method F. The FDS-ANSYS and JASMINE-STELAb couplings use Method E with a slight modification: a separate code calculates the incident radiation flux on each mesh-node j based on the directional radiance values supplied by the CFD code and the geometric data supplied by the solid-phase code. For convex elements the ANSYS code takes into account the radiative heat transfer between parts of the solid which see each other due to surface emission. The VESTA-DIANA coupling is based on the two-way coupling Method B. The structural elements are modelled in the FEM code as FEM solids and in the CFD code as CFD solids. The CFD solids envelope the FEM solids, the latter being modelled in more detail. All interpolations and radiation calculations are performed within the CFD code VESTA.

For use in one-way Methods C – F, a generic interface file specification has been established. The structure of the file is specified below via a hypothetical example (Table 2). It is given for the general scenario which enables the possibility of the integration of the flux to be made in the CFD analysis, which is the preferred case because it allows for a better treatment of the concave elements. NF or NI can be given the value of 0, if required, where information about the flux and the radiance values, respectively, is not present in the file.

Table 2. Proposed generic interface file format for one-way coupling [6].

FILENAME cfd.txt	Gives the name of the file.
NSTEPS 6	Gives the number of time-steps in the file.
TIMES 30. test_example_30s.simd 60. test_example_60s.simd ... 600. test_example_600s.simd	Lists the times (s) when the heat fluxes and/or radiance values are given, optional name of the files after each time.
NF 288	Gives the number of transfer points where the flux is given (NF points).
XYZ_FLUXES 10.805 14.8 1. 0. -1. 0. 10.9 14.8 1. 0. -1. 0. ...	NF lines, each containing the X, Y, Z location (m) of a transfer point and the components (X, Y, Z) of the outward normal to the surface in the Global system of coordinate.
NP 144	Gives the number of transfer points (NP points) where the radiance values are given.
XYZ_INTENSITIES 10.7 14.72 1.	NP lines. Each line gives the location X, Y, Z of the transfer point where the radiance values ($W/m^2/sr$) are given.

10.9 14.72 1. ...	
NI_DIRECTION 20	Gives the number of radiance directions (NI).
XYZ_DIRECTIONS 0.0000 0.3568 0.9342 0.0000 -0.3568 -0.9342 ...	NI lines. Each line contains X, Y, Z component of a radiance direction viewed from point 'outwards'.
TIME 30	Gives the time-step (s) where the subsequent values of heat fluxes and radiance values are given. This block and the two blocks that follow FLUXES and INTENSITIES are repeated NSTEPS time.
FLUXES 300. 10. 2500. 300. 10. 2500. ...	NF lines. Each line contains gas temperature in K, convection heat transfer coefficient ($W/m^2/K$) and incident radiative flux (W/m^2).
INTENSITIES 300. 10. 125. 760. ... 125. 300. 10. 125. 760. ... 125. ...	NP lines. Each line contains gas temperature in K, convection heat transfer coefficient ($W/m^2/K$) and NI radiance values ($W/m^2/sr$) [integrated over solid angle to give radiant intensities (W/m^2)].

The file is not formatted, which means that any format can be used to write data in the file, for example 3.5E+03 as well as 3500. Each data in a record has to be separated from the other data by a separator, preferably one (or several) blank character(s). At the beginning of each block of relevant information is a heading (keyword) in capital letters. It is permitted to add a comment after the heading, with a space in between, but the comment must be confined to the same line.

The utility of such a file is in facilitating extension of coupling to include codes not yet considered; a one-way coupling linkage can therefore be established purely by developing new program code to exchange information with this file, either by writing out (CFD) or reading in (solid phase).

VALIDATION

A systematic testing of selected one-way and two-way coupling methodologies within the integrated framework was been carried out using the coupled codes, for three main test cases: a hypothetical benchmark scenario with localised fires and various steel components set in a large compartment [6], an experimental set-up involving a loaded beam-column assembly subjected to a flashed-over fire in a smaller compartment (CTICM external column fire tests [7]), and a large open car-park structure exposed to transient car fires under varying wind conditions [8]. Steel was unprotected, excepting the CTICM beam.

Benchmark localised fire case [6]

A hypothetical scenario was defined involving a selection of exposed steel columns and beams (see Fig. 6). Calculations were made for the thermal response of various steel tubes and I-profile columns and beams inside a compartment 30 m by 20 m in floor area and 10 m high, open at the end remote from the fire (2.5m soffit). The tubes were hollow, rectangular steel sections (hence no shadow effect), and steel I profile sections (and hence including a shadow effect). Details of the different steel sections studied are given in Table 3; in each case, a uniform section thickness of 1mm was assumed, in order to effectively eliminate the transient heating effects. The fuel was heptane, with an assumed source soot yield (mass fraction) of 0.088. Fast-growing t^2 fires were specified in 5m square trays at one of two locations (A/B), reaching 30 MW in 800s and maintained at this level thereafter, such that a hot smoke layer was generated under the entire ceiling, with a layer of clear air below. This allowed the performance of the different coupling methodologies to be examined and compared in respect to both beams, which are completely immersed in a hot ceiling layer, and columns, which are partly engulfed in the hot layer but heated additionally lower down by direct radiation from both the fire and the smoke layer above. Adiabatic surfaces were defined for the compartment boundaries so that reflected radiation was also a potential mechanism for heat transfer. Eurocode properties were assumed for the steel, and the surface emissivity was taken to be 0.7.

Table 3. Steel section details for benchmark scenario [6].

Element type	Short beam (20m)	Long beam (30m)	Columns (10m)
Hollow tube	1000 x 400 mm	500 x 200mm	400 mm square
I-section	IPE 500 profile	HLB 100+ profile	HD 400 x 314 profile

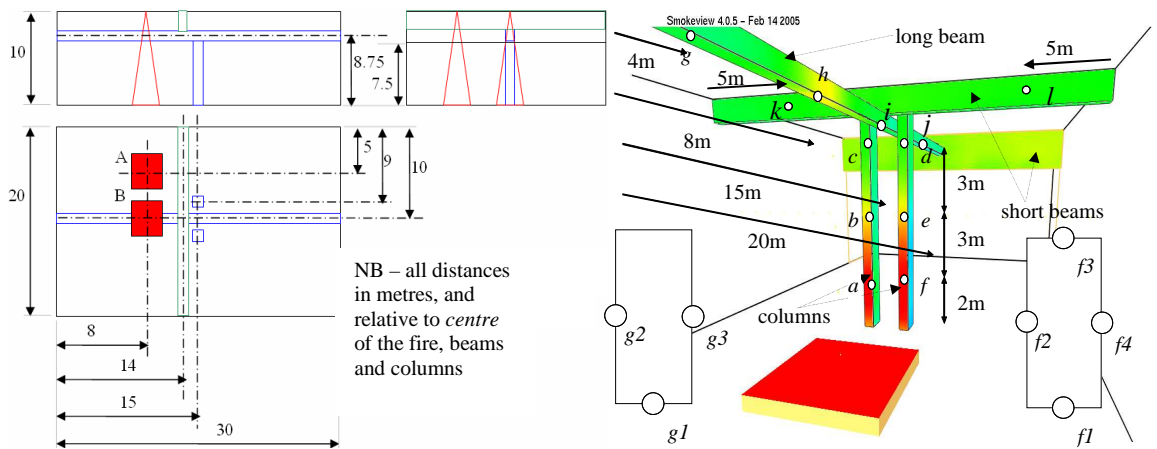


Fig. 6. Benchmark case geometry: a) elevations and plan, b) isometric, with tube plan/cross-sections [6].

Calculations were performed for each of the code pairs mentioned in Table 1. Selected results are shown in Fig. 7 for locations within and below the hot layer, respectively, for the case with the fire at location A and with all steel components being hollow tubes. Whilst both one- and two-way methods give generally plausible results, some obvious differences are apparent in the steel temperature predictions. In the hot layer, Fig. 7 a), at locations $g3$ and $l3$, the differences are fairly small, with the two-way coupling temperatures exceeding those of one-way coupling and in each case the location g temperature, nearer the fire, being higher, as expected. The reasons for the differences between code pairs are mainly attributable to variations in the underlying gas-phase temperature predictions and radiation fields.

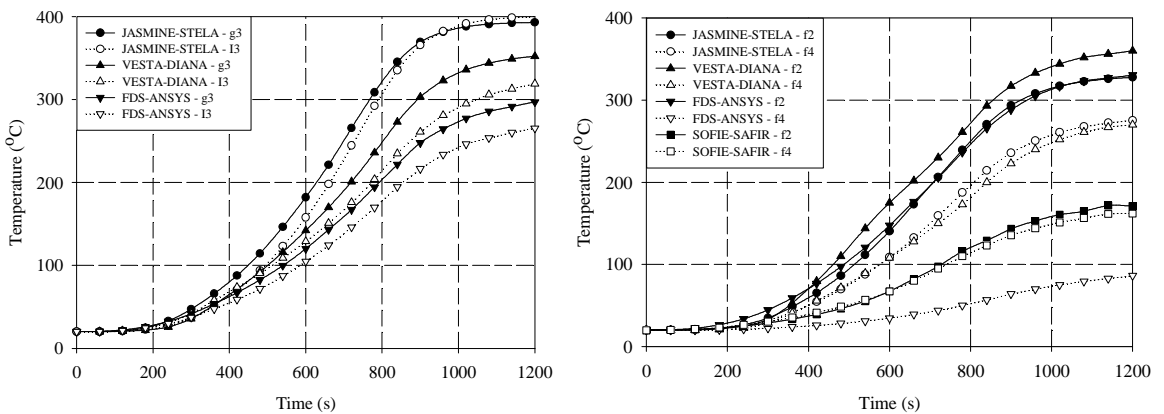


Fig. 7. Example of predicted steel temperatures, a) in hot layer b) below hot layer, for different code pairs.

In the cold layer, Fig. 7 b), at locations $f2$ and $f4$, bigger differences are observed. Here, the two-way methods again predict higher steel temperatures, and the results for the more exposed location, $f2$, are generally higher than those at $f4$, in-line with expectations. However, the sensitivity to this exposure difference varies greatly between code pairs, being very high for one-way FDS-ANSYS coupling, but quite low for one-way SOFIE-SAFIR coupling (very similar results were also obtained with the other Method F pairings, JASMINE-SAFIR, VESTA-SAFIR). The reason for the discrepancy may lie either in the redistribution of the energy within the solid section via the thermal model or, more likely, in differences in outputs derived from the CFD code radiation (post-processing) calculation. The respective one-way methods are distinguished by a fundamental difference in this treatment, with incident radiative fluxes to solid surfaces computed directly by the CFD-radiation model in Method E but, more ambitiously, a field of directional radiance values passed on a mesh of sampling points in Method F. The former approach may generally be more robust, but with the latter data storage memory is saved.

Finally, the effect of the resolution of the directional information was examined, with 100 rays found to give acceptable results for all pairs, but 10 to 20 rays being insufficient, with important near-field effects missed. This is not a problem in general, since the post-processing calculations are fairly modest.

CTICM external column fire case [7]

A series of tests was performed in the 1970's at CTICM's Maizières-les-Metz test laboratory, to determine the fire-resistance of loaded steel beam-column assemblies subjected to heating from a wood crib fire in an adjacent compartment. Test 13 was selected for the validation of the full one-way coupling methodology. Predictions of peak gas-phase temperatures around the column were within 50°C of the measurements, and the time history trend well reproduced, using FDS. These temperatures were modified by less the 10°C when the structural component was omitted from the CFD model. The coupled prediction of deflections, using ANSYS via Method E, also reproduced the measured trends very well, with a peak value of 9mm, cf. 10mm measured for the column, and corresponding values of 9mm and 15mm for the beam [5,6].

Open car-park case [8]

Test 2 was examined in detail. It was first established that the fire temperature predictions of FDS and JASMINE were sufficiently close to the measurements. In JASMINE, the effect of using either adiabatic or isothermal blocks to represent the beams was investigated, with peak temperatures near the cars being up to 170°C higher for the former case, and the two results spanning the measured value. The steel temperatures obtained via one-way coupling to ANSYS and STELA, using Methods E and F respectively, verified the sufficiency of the thermal couplings. The effect of omitting the steel beams entirely from the CFD was then examined and found to be very significant, as expected, because of the impact on the ceiling layer resulting from the complete lack of confinement [6]. Inclusion of the beams produced more conservative results.

DISCUSSION

In general, for scenarios such as the benchmark case and CTICM external fire test, where the beams and columns which do not interfere strongly with the flow-field, one-way coupling is found to produce results which are equally satisfactory to those of the much more demanding two-way coupling, even when the component itself is omitted from the calculation. This simple one-way coupling is more computationally efficient – it permits a single CFD simulation to be used for many subsequent structural analysis calculations – and it also allows the CFD and structural modelling to be performed remotely by experts in their respective disciplines, i.e. fire simulations can be executed completely independent of the thermal response model, and *vice versa*. Further, the geometrical model of (parts of) the structural elements in the fire development model is liberated to adopt an independent, generally much coarser, spatial scale than that specified in the structural model. However, there are further considerations as follows.

An obvious drawback of the one-way approach is that some degree of error may be introduced in the fire development model due to the inaccurate representation of the heat lost by transfer into the structural element, particularly when the structural component is omitted entirely in the fire simulation. The magnitude of this error is very scenario specific, and in the external column fire case examined here the effect was found to be very small, but much larger for the car park scenario. A distinction must also be made between the thermal modelling of enclosure boundaries and individual structural elements of interest. For the former, there is generally no alternative to undertaking a two-way coupling, otherwise errors will arise in the predicted gas-phase temperatures. This is already included in CFD fire models, where most often a simple one-dimensional treatment in the solid suffices for enclosure boundaries. Thus, in cases where structural components are large, their thermal effects must also be considered via two-way analysis.

Another issue, potentially even more significant, is the impact of the structural components on the flowfield itself. This is a particularly important factor when the structural components are omitted from the flowfield calculations entirely, if they provide a significant obstruction, e.g. beams which confine the movement of hot fire gases under the ceiling. The car park simulations showed how important this effect can be. Hence, where this may be the case, consideration should be given to including the main blockages in the CFD calculation, as simplified (and optionally, adiabatic) versions of the actual elements, with the thermal response of the structural elements subsequently calculated by a one-way coupling approach. Flowfield effects may also be very important at a local scale, since the adoption of simplified component geometries impacts both radiative and convective heat transfers via the compromises made in representing the details of the geometrical boundaries of the element. Exchange of directional radiation information may help to overcome this problem, but it must be recognized that radiative exchange between the surfaces of the structural component is still omitted and convection will necessarily be highly approximated, at best.

CONCLUSIONS

A hierarchy of coupling methodologies for integrating the CFD and solid-phase (thermal and structural response) models has been established and tested. The methods differ in the type of data exchanged and thus in the kind of conversion performed by each code, and the consequent computational demands. The most fundamental division is between one-way methods, where information flows only from the CFD code to the solid-phase code, and two-way methods where information is returned to the fire model, representing “loose” and “tight” coupling respectively. The latter requires interactive performance of the calculations, and is much more computationally demanding, but does confer a (scenario-specific) accuracy advantage. Within each category, further distinctions can be made according to choices about which parameters to pass between the codes, e.g. whether to pass heat fluxes computed for objects represented directly in the CFD code, or, instead to pass (sampled) fields of directional radiance information. Again, these choices affect the balance of effort required in each code, both in terms of computational costs and complexity, the flexibility of the procedure, e.g. in determining whether or not a single set of CFD results can be used for a variety of solid-phase models, and, potentially, on the accuracy of the results.

The different coupling methodologies have been verified in a variety of applications and found to perform generally adequately. Differences between the codes were attributed mainly to the details of the CFD treatments, and radiation in particular, and not to the code coupling mechanism itself. For most cases, the errors introduced by the compromises required for one-way coupling are not found to be unreasonably large and the flexibility of this approach would often be advantageous, particularly where omission of structural components from the fire model can be justified. However, where structural components can significantly affect the computed flowfields there is generally no alternative to representing their boundaries in the fire model, at least approximately, though adiabatic blockages may often suffice.

ACKNOWLEDGEMENTS

We acknowledge the significant contributions to this work made by the other members of the FIRESTRUC consortium, including Joris Fellingner and Leander Noordijk (TNO, now Efectis), Jean-Marc Franssen and Dan Pintea (University of Liège), Bin Zhao, Antoine Breuillard, Julian Darche and Sylvain Desanghere (CTICM), and Olivier Vassart and Louis-Guy Cajot (ProfilARBED-Research, now ArcelorMittal).

REFERENCES

- [1] Fellingner, J.H.H., Kenyon, Y.M., Breunese, A.J. and Lemaire, A.D. 2004. A coupled CFD-FEA study of a tunnel structure exposed to a fire, INTERFLAM 10: 1349–1360.
- [2] Prasad, K. and Baum, H.R. 2005. Coupled fire dynamics and thermal response of complex building structures. Combustion Symposium 30: 2255–2262. doi:10.1016/j.proci.2004.08.118
- [3] Baum, H. 2005 Simulating fire effects on complex building structures, Fire Safety Science 8: 3–20. doi:10.3801/IAFSS.FSS.8-3
- [4] Jowsey, A., Rein, G., Abecassis-Empis, C., Cowlard, A., Reszka, P., Steinhaus, T., Torero, J.L., Usmani, A. and Lane, B. 2008. An analytical approach to define surface heat fluxes to structural members in post-flashover fires. Fire & Explosion Hazards 5, Edinburgh, UK, 23–27 April, 2007.
- [5] Zhao, B., Desanghere, S. and Darche, J. 2007. CFD and FEM coupling with computer programs FDS and ANSYS for fire resistance assessment. INTERFLAM 11: 1461–1466.
- [6] Kumar, S., Miles, S., Welch, S., Vassart, O., Zhao, B., Lemaire, A.D., Noordijk, L.M., Fellingner, J.H.H. and Franssen, J.-M. 2008. FIRESTRUC – Integrating advanced three-dimensional modelling methodologies for predicting thermo-mechanical behaviour of steel and composite structures subjected to natural fires, RFCS project RFS-PR-02110, European Commission report.
- [7] Kruppa, J. 1981 Some results on the fire behaviour of external steel columns, Fire Safety Journal 4: 247–257. doi:10.1016/0379-7112(81)90028-X
- [8] Joyeux, D., Kruppa, J., Cajot, L.-G., Schleich, J.-B., Van de Leur, P. and Twilt, L. 2002. “Demonstration of real fire tests in car parks and high buildings”, ECSC project 7215-PP-025, European Commission report.